



Introduction:

Designed for high current drive with high flux output and presented in a compact package outline (6.0 x 6.0 x 1.5 mm), these SPNova products showcase the latest technological advent in its range. With its ultra low height profile, 1.5mm, SPNova presents a unique combination of compactness and unbounded brightness.

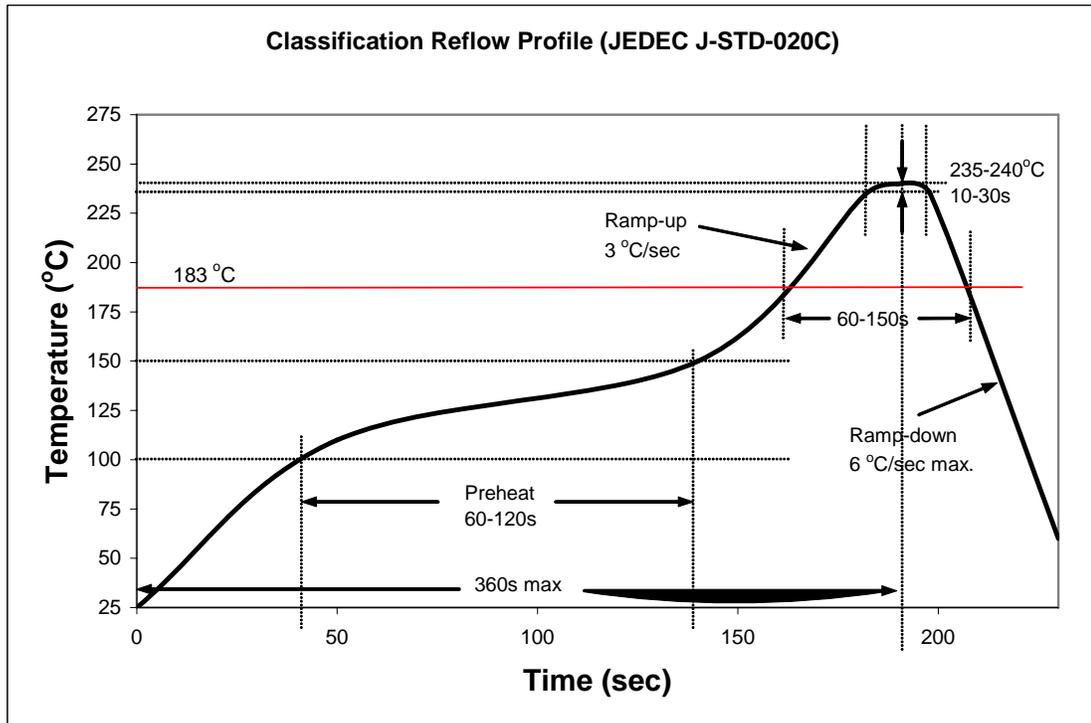
Standard Soldering Process:

The SPNova soldering surfaces are plated with 100% tin (Sn) and are therefore RoHs compliant. The component is designed to be compatible to the existing industry SMT process and IR-reflow. There are no special processes or equipment required for the mounting of the components onto applications. Both the thermal and electrical connections are provided by the conventional process. Therefore, there is no need to provide for additional process or material to take care for the thermal connection.

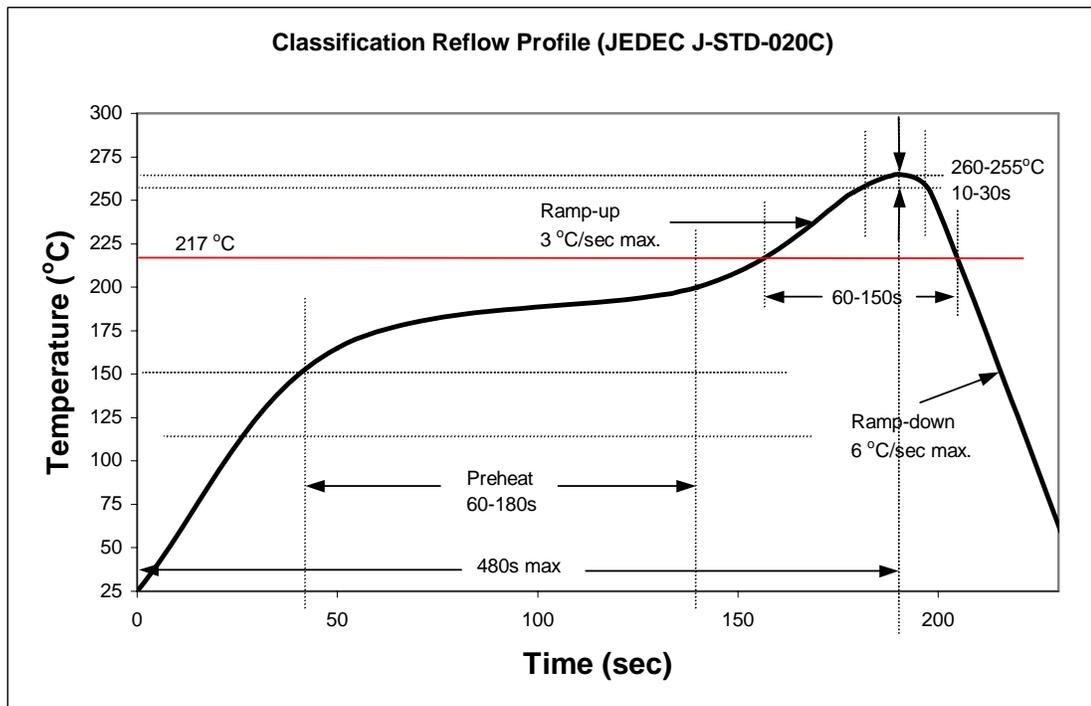
However, due to the unique design, all the soldering terminals are located at the bottom surface of the component. This greatly reduces the space required and also enhances the thermal dissipation capability of the component. Heat from the LED chip is directly conducted via the soldering terminals to the external environment. Thermal path is kept to the very minimum.

As for the soldering process, the component is qualified for both Pb and Pb-free soldering profile. Both the profiles are as per described in the datasheet.

Recommended Sn-Pb IR-Reflow Soldering Profile.



Recommended Pb Free IR-Reflow Soldering Profile.



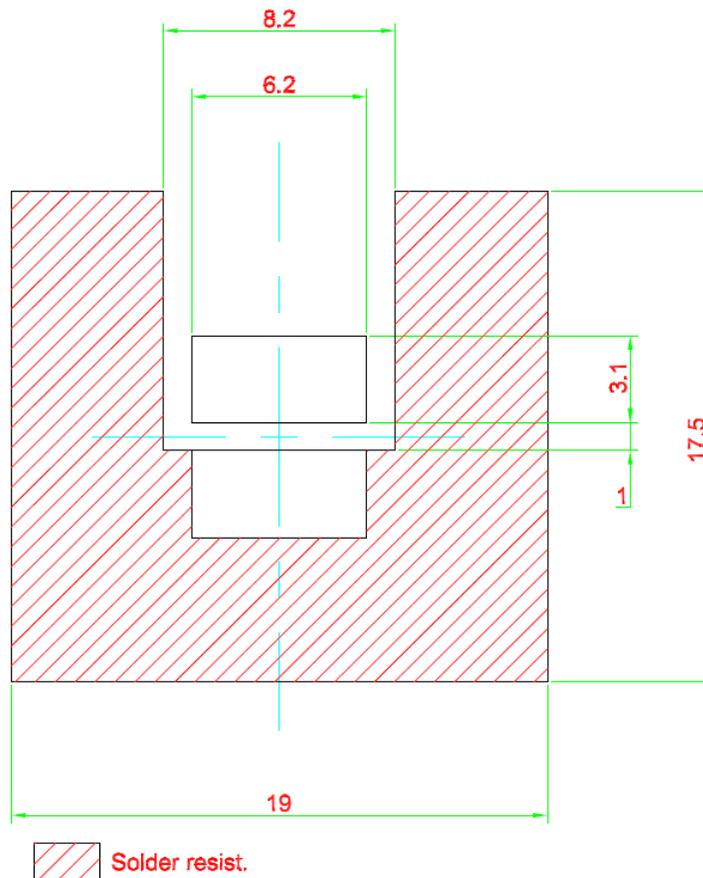
Surface Mounting – Factors to Consider:

This application note provides a guideline for the surface mounting of SPNova. The following parameters have to be considered in order to optimize the surface mounting performance.

- > Solder pad size
- > Solder stencil size
- > Pipette (nozzle)
- > Solder paste thickness

Solder pad size

The recommended solder pad design is as illustrated in the data-sheet. It may differ depending on the device. A single chip device such as SPNova White (NPW-TSD) would have a pad design as illustrated below.



A typical recommended solder pad design.

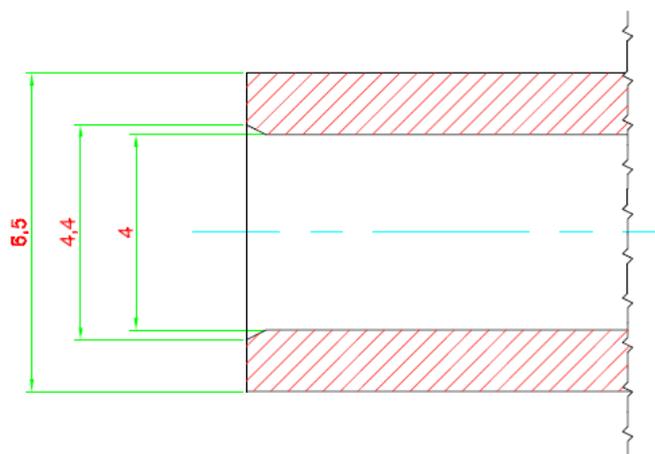
Solder Stencil

In order to minimize solder bridging problems, it is common to design stencil aperture size smaller than the recommended solder pad. Excessive amount of solder paste deployed will result to tilted parts and inaccurate placement position. This is particularly important for the SPNova RGB where the gap between the pads is small.

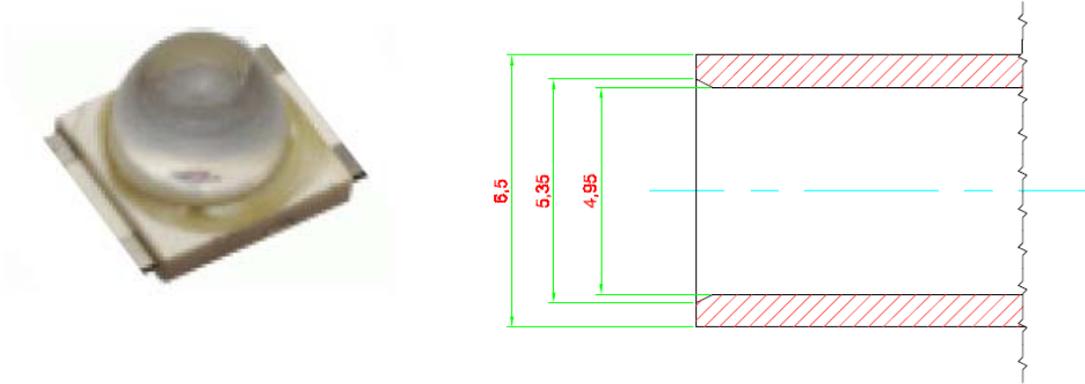
It is recommended that the aperture is reduced to 75% of the recommended solder pad design.

Pipette

Pick and place machine should be able to process SPNova devices with the required placement accuracy. Care should be observed that the surface of the pipette which is in contact with the LED is flat and smooth. The amount of force exerted during pick-up and landing should also be minimized. Pipette design suitable for SPNova is as depicted in the photo below. Anyway depending on the type of SMT machine use, the pipette design might vary from our recommendation. As a rule of thumb, the pipette dimension use must avoid contacting LED emitting area. Excessive force exerted on the LED emitting surface may damage the part as the cavity is filled with soft encapsulant.



Standard SPNova package with recommended pipette dimension

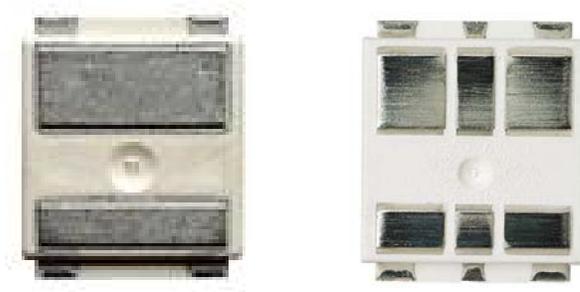


SPNova with lens package with recommended pipette dimension

Solder paste thickness

We recommend using minimum solder paste in order to achieve a good solder formation. A solder paste thickness of 0.125 mm will be optimum.

For SPNova components, no solder fillet will be observed after re-flow process as the soldering surfaces are at the bottom.



Soldering surfaces are located on the bottom of the components.